

**DRAFT 4/97**

## Chapter II: Software Design

## 1.0 PURPOSE

In this chapter, the FEHM software is described, including functional requirements, input/output files and their associated variables, and error and warning conditions with their associated messages.

## 2.0 FUNCTIONAL DESCRIPTION

After a user formulates a problem to be modeled, a mesh representing the physical media is generated. It should be noted that mesh generation is not a primary function of the FEHM application and auxilliary mesh-generation programs are generally used to obtain this input. Data describing the mesh and physical properties of the media and the fluid/gas system to be modeled are input. The appropriate finite-element equations are formulated and constitutive relationships applied. The solution is then computed, and results and restart data are output.

## 3.0 ASSUMPTIONS AND LIMITATIONS

The assumptions and limitations inherent in FEHM can be categorized as follows:

- The validity of the underlying partial differential equations used to describe the physics of the coupled processes of heat and mass flow in a porous media (see Zyvoloski et al. 1997a). This limitation includes the difficulty in obtaining appropriate input parameters to model given experimental and field data.
- The validity of the computer code used to solve the partial differential equations that describe the physics of flow. This assumption is described further in the last two chapters: "Verification and Validation Plan" and "Verification and Validation Report."
- The ability of the computer code to obtain the advertised performance. This limitation is both machine dependent and user dependent. FEHM is state of the art in both numerical performance and memory management. The numerical performance allows very large (100,000-node) 3-D problems to be run in a workstation environment. Thus, the numerical accuracy needed for typically fine-resolution problems such as contaminant transport can be realized. The memory management allows many jobs to be run on a single workstation and allows quick analysis of problem sensitivities. In our experience, the largest source of poor code performance, both in accuracy and CPU speed, is user error. FEHM is a complicated tool and requires a high level of user experience in simulation and groundwater applications. The FEHM "User's Manual" and other documents (Zyvoloski et al. 1997b) are updated regularly with major input from users.

## 4.0 PRIMARY PRODUCT DESCRIPTIONS

The FEHM application consists of one primary product, program **fehm**, which is used to formulate a transient problem that generates a set of nonlinear algebraic equations. The nonlinear equations are solved using the GZSOLVE application (Zyvoloski and Robinson 1995).

### 4.1 FEHM

#### 4.1.1 Purpose

See Section 1.0 of this chapter.

The functional requirements satisfied by the **fehm** program are summarized in Table 1, where the section numbers refer to the previous chapter, "Software Requirements Specification."

**DRAFT 4/97**

**Table 1. Functional requirements of the FEHM application**

Requirement	Section	Implemented by
<b>Finite-element coefficient generation</b>	3.1	anomp, area2d_tri, check_sx, determ, gencof, gncf2, gncf3, mdnodes, pebi, pebi3, radius, setzone, sfn2r, sfn3r, shap2r, shap3p, shap3r, split, sx_combine, thickness
<b>Formulate transient equations</b>	3.2	
Heat-conduction equations	3.2.1	bnsver, geneq3, gensl3
Heat- and mass-transfer equations	3.2.2	air_rdof, aircrt, bnsver, co2ctr, dpdp3, dpdpfa, dpdpfh, dualex, dualfa, dualfh, geneq1, geneq2, geneqc, geneqmdnode, gensdp, gensdp3, gensl1, gensl2, gensl4, interblock_iso, mod_eqs_ngas, thermw, thrair, thrmwc, varchk, zeolites
Noncondensable-gas flow equations	3.2.3	air_rdof, aircrt, bnsver, co2ctr, dualfa, geneq2, geneqc, geneqmdnode, gensdp, gensl2, gensl4, interblock_iso, thrair, thrmwc, varchk
Solute-transport equations	3.2.4	angle3, cnswer, concadiff, concen, coneq1, coneq1mdnode, csolve, dppta, dualta, dualtx, eullag3, gencon, gentdp, node_rxn, react, rotate, solstore, thermc, varchk
Particle-tracking module	3.2.5	cell_time, getconc, inptrk, inverf, part_track, set_ptrk, time_diff, time_disp2, wrptrk
Sources and sinks	3.2.6	bcon, infl02, inflow, inhflx, thermw, thermc, thrair, thrmwc, zeolites
<b>Apply constitutive relationships</b>	3.3	
Pressure- and temperature-dependent water properties	3.3.1	data, enthp, outbnd, psat, psatl, thermw, thrmwc, thrair, varchk
Properties of air and air/water vapor mixtures	3.3.2	air_cp, dvacalc, outbnd, psatl, thrair, thrmwc, vaporl, varchk
Equation-of-state models	3.3.3	coeffc, sther, varchk
Relative-permeability and capillary-pressure functions	3.3.4	cappr, rlperm, vgcap, vgcap_fit, vgrlp
Adsorbing solutes	3.3.5	mult_rxn, rdcon, read_rxn, solstore, thermc
Multiple, interacting solutes	3.3.6	mult_rxn, node_rxn, react, read_rxn, rxn_product, setup_rxn, thermc
Dual-porosity formulation	3.3.7	dual, dualex, dualfa, dualfh, dualta, dualtx
Double-porosity/double-permeability formulation	3.3.8	crdpdp, ctddpd, dpdp, dpdp3, dpdpfa, dpdpfh, dppta, indpdp, rddpdp
Stress-dependent properties	3.3.9	porosi, vfcal
Variable thermal conductivity	3.3.10	vcon

**Table 1. Functional requirements of the FEHM application (continued)**

Requirement	Section	Implemented by
<b>Compute solution to transient equations</b>	3.4	
Implement time-step mechanism	3.4.1	csolve, daycrl, fehmn, intime, resetrc, resetv, timcrl
Solve nonlinear equation set at each time step	3.4.2	bnsver, cnswer, gencon, gensdp, gensdp3, gensl1, gensl2, gensl3, gensl4, gentdp, mod_eqs_ngas, normal, normal_dof, nrmlz4, rd1dof, rdof_new, setord, solve, switch, switchb, and GZSOLVE application (Zyvoloski and Robinson 1995)
<b>Provide input/output data files</b>	3.5	avs_io, cntlin, cntlio, done_macro, file_prefix, infiles, iofile, namefile2, setunits, start_macro, termin, termio, writeio
Inputs	3.5.2	airctr, co2ctr, concen, disk, diskc, diskp, dpdp, dual, flxo, gendat, geoin, incond, incoord, inctrl, inflo2, inflow, inhflx, initdata, inmentat, innode, inpatran, inperm, inpres, inptrk, input, inrock, intime, inzeol, mdnodes, porosi, rdcon, rddpd, rdthick, read_avs_io, read_rxn, read_sx, renun, rlperm, scanin, setparams, sice, sther, storsx, user, vcon, welbor, zone
Outputs	3.5.4	avs_write_cord, avs_write_struc, contr, contrc, contrj, datchk, diagnostics, disk, diskc, diskp, elem_type, elem_type_binary, flxo, input, outbnd, plot, plotc1, storsx, write_avs_node_con, write_avs_node_mat, write_avs_node_s, write_avs_node_v, write_avs_ucd_header, write_binary_geo, write_binary_header, write_binary_node_con, write_binary_node_mat, write_binary_node_s, write_binary_node_v, wrtcon, wrtout, wrptrk, zone
<b>Provide restart capability</b>	3.6	
Write information needed for a restart to an output file	3.6.1	disk, diskc, diskp, timcrl
Read information needed for a restart from a restart file	3.6.2	disk, diskc, diskp, startup
Resume the calculation	3.6.3	concen, fehmn, startup

#### 4.1.2 Functional description

See Section 2.0 of this chapter.

A simplified flow chart illustrating the segmentation of the application is provided in the FEHM document “Summary of Models and Methods” (Zyvoloski et al 1997a). A summary listing of the subroutines and functions invoked by the **fehm** program is given in Appendix A of this report.

**DRAFT 4/97****4.1.3 Assumptions and limitations**

See Section 3.0 of this chapter.

**4.1.4 Input/output****4.1.4.1 Configuration interface table**

N/A

**4.1.4.2 Other input data files**

**4.1.4.2.1 Input control file.** The input control file provides the **fehm** program with the names of input/output (I/O) files to be used by the program, the terminal output control flag, and the user subroutine number (Table 2). Names for the files, which may include subdirectories or paths, are entered one per line followed by the terminal output control flag and user subroutine number. A blank line may be entered for any item not required by the user. Variable *nmfil(1)* contains the name of the input control file. The default control file name is *fehmn.files*.

**Table 2. Input control file**

Input variable	Format	Description
<i>nmfil( 2)</i>	character*100	Main input file name
<i>nmfil( 3)</i>	character*100	Geometry-data input file name
<i>nmfil( 4)</i>	character*100	Zone-data input file name
<i>nmfil( 5)</i>	character*100	Main output file name
<i>nmfil( 6)</i>	character*100	Restart input file name
<i>nmfil( 7)</i>	character*100	Restart output file name
<i>nmfil( 8)</i>	character*100	Simulation-history output file name
<i>nmfil( 9)</i>	character*100	Solute-history output file name
<i>nmfil(10)</i>	character*100	Contour-plot output file name
<i>nmfil(11)</i>	character*100	Dual-porosity or double-porosity/double-permeability contour plot output file name
<i>nmfil(12)</i>	character*100	Coefficient-storage file name
<i>nmfil(13)</i>	character*100	Input-check output file name
<i>tty_flag</i>	character*4	Terminal output flag: all, some, none
<i>usub_num</i>	integer	User subroutine call number

**4.1.4.2.2 Main input data file.** The main input file contains user parameter initialization values and problem control information (Table 3). The default input data file name is *fehmn.dat*.

## DRAFT 4/97

**Table 3. Main input data file**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>wdd</i>	character*80	<i>Title line.</i>
<i>macro</i>	character*4	Any macro keyword described below, except <b>coor</b> and <b>elem</b> , may use <i>lockeyword</i> and <i>locfilename</i> to designate that the regular data should be read from an auxiliary input file.
<i>lockeyword</i>	character*4	Keyword “ <b>file</b> ” to designate that an auxiliary input file is used.
<i>locfilename</i>	character*100	Name of the optional data input file.
<i>macro</i>	character*4	Keyword “ <b>adif</b> ” (optional).
<i>tort</i>	real	Tortuosity for air-water vapor diffusion.
<i>macro</i>	character*4	Keyword “ <b>airwater</b> ” (optional).
<i>ico2d</i>	integer	Type of air module to use.
<i>tref</i>	real	Reference temperature for properties (°C).
<i>pref</i>	real	Reference pressure for properties (MPa).
<i>macro</i>	character*4	Keyword “ <b>alti</b> ” (not supported this version).
<i>macro</i>	character*4	Keyword “ <b>bous</b> ” (optional).
<i>icons</i>	real	Any nonzero integer to enable Boussinesq-type approximation.
<i>macro</i>	character*4	Keyword “ <b>cond</b> ” (required).
<i>ja</i>	integer	First node to be assigned properties. If <i>ja</i> < 0, the zone to be assigned properties.
<i>jb</i>	integer	Last node to be assigned properties.
<i>jc</i>	integer	Loop increment for assigning properties.
<i>thxd</i>	real	Thermal conductivities in the x-direction, y-direction, and z-direction
<i>thyd</i>	real	
<i>thzd</i>	real	$\left( \frac{W}{m \cdot K} \right)$ .
A blank line to signal end of conductivity input.		
<i>macro</i>	character*4	Keyword “ <b>cont</b> ” (optional).
<i>ncntr</i>	integer	Time-step interval for contour plots (number of time steps).
<i>contim</i>	real	Time interval for contour plots (days).
or		
<i>altc</i>	character*4	Keyword specifying the type of contour output wanted: “ <b>avs</b> ”, “ <b>fehm</b> ”, “ <b>free</b> ”, “ <b>ment</b> ”, “ <b>pctr</b> ”.
<i>ncntr</i>	integer	Same as above.
<i>contim</i>	real	Same as above.
<i>chdum</i>	character*72	Keyword used when <i>altc</i> is “ <b>avs</b> ” specifying type of contour-plot data files to be created. Valid keywords (case insensitive) are: (m)aterial - output contour values for material properties. (l)iquid - output contour values for liquid phase. (v)apor - output contour values for vapor phase. (v)e locity - output velocity values. (dp)dp - output contour values for dual-permeability nodes. (p)ressure - output pressure values. (t)emperature - output temperature values. (s)aturation - output saturation values. (c)oncentration - output solute concentration values. (f)ormatted - output data in ASCII format.

## DRAFT 4/97

**Table 3. Main input data file (continued)**

Input variable	Format	Description
		(u)nfomatted - output data in binary format. (e)ndavs - last keyword entered. Keywords are entered one per line, and the letters given in ( ) are sufficient to identify the keyword.
<i>macro</i>	character*4	Keyword “ <b>coor</b> ” (required). See Section 4.1.4.2.3 on page 43.
<i>macro</i>	character*4	Keyword “ <b>ctrl</b> ” (required).
<i>maxit</i>	integer	Maximum number of iterations allowed in either the overall Newton cycle or the inner cycle to solve for the corrections at each iteration.
<i>epm</i>	real	Tolerance for Newton cycle (nonlinear-equation tolerance).
<i>north</i>	integer	Number of orthogonalizations in the linear equation solver.
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>igaus</i>	integer	The order of partial Gauss elimination. Larger values increase memory use but may be necessary for convergence.
A blank line to signal end of Gauss elimination order input.		
<i>aaw</i>	real	Implicitness factor.
<i>agrav</i>	integer	Direction of gravity.
<i>upwgt</i>	real	Value of upstream weighting.
<i>iamm</i>	integer	Maximum number of iterations for which the code will multiply the time-step size.
<i>aiaa</i>	real	Time-step multiplier.
<i>daymin</i>	real	Minimum time-step size (days).
<i>daymax</i>	real	Maximum time-step size (days).
<i>icnl</i>	integer	Parameter that specifies the geometry.
<i>lida</i>	integer	Parameter that specifies the external storage of geometric coefficients.
<i>macro</i>	character*4	Keyword “ <b>dof</b> ” (not implemented).
<i>macro</i>	character*4	Keyword “ <b>dpdp</b> ” (optional).
<i>idpdp</i>	integer	Solution descriptor for double-porosity/double-permeability solution.
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>volfd1</i>	real	Volume fraction for fracture node.
A blank line to signal end of volume-fraction input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>apuv1</i>	real	Length scale for matrix nodes (m).
A blank line to signal end of length-scale input.		
<i>macro</i>	character*4	Keyword “ <b>dual</b> ” (optional).
<i>idualp</i>	integer	Solution descriptor for dual-porosity solution.
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>volfd1</i>	real	Volume fraction for fracture portion of the continuum.
A blank line to signal end of volume fraction for fracture input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>volfd2</i>	real	Volume fraction for the first matrix portion of the continuum.
A blank line to signal end of volume fraction for matrix input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>apuvd</i>	real	Length scale for matrix nodes (m).
A blank line to signal end of length-scale input.		
<i>macro</i>	character*4	Keyword “ <b>elem</b> ” (required). See Section 4.1.4.2.3 on page 43.

## DRAFT 4/97

**Table 3. Main input data file (continued)**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>	character*4	Keyword “eos” (optional).
<i>ieosd</i>	integer	Equation-of-state reference number.
<i>ipsat</i>	integer	Parameter to set vapor pressure to zero.
<i>itsat</i>	integer	Parameter to adjust the saturation temperature.
<i>ew1</i>	real	Liquid reference pressure (MPa).
<i>ew2</i>	real	Liquid reference temperature (°C).
<i>ew3</i>	real	Liquid reference density (kg/m <sup>3</sup> ).
<i>ew4</i>	real	Derivative of liquid density with respect to pressure at reference conditions.
<i>ew5</i>	real	Derivative of liquid density with respect to temperature at reference conditions.
<i>ew6</i>	real	Liquid reference enthalpy (MJ/kg).
<i>ew7</i>	real	Derivative of liquid enthalpy with respect to pressure at reference conditions.
<i>ew8</i>	real	Derivative of liquid enthalpy with respect to temperature at reference conditions.
<i>ew9</i>	real	Liquid reference viscosity.
<i>ew10</i>	real	Derivative of liquid viscosity with respect to pressure at reference conditions.
<i>ew11</i>	real	Derivative of liquid viscosity with respect to temperature at reference conditions.
<i>ev1</i>	real	Vapor reference pressure (MPa).
<i>ev2</i>	real	Vapor reference temperature (°C).
<i>ev3</i>	real	Vapor reference density (kg/m <sup>3</sup> ).
<i>ev4</i>	real	Not used, included only to maintain a similar format to liquid parameters.
<i>ev5</i>	real	Not used, included only to maintain a similar format to liquid parameters.
<i>ev6</i>	real	Vapor reference enthalpy (MJ/kg).
<i>ev7</i>	real	Derivative of vapor enthalpy with respect to pressure at reference conditions.
<i>ev8</i>	real	Derivative of vapor enthalpy with respect to temperature at reference conditions.
<i>ev9</i>	real	Vapor reference viscosity.
<i>ev10</i>	real	Derivative of vapor viscosity with respect to pressure at reference conditions.
<i>ev11</i>	real	Derivative of vapor viscosity with respect to temperature at reference conditions.
<i>macro</i>	character*4	Keyword “finv” (optional; no input is associated with this macro).
<i>macro</i>	character*4	Keyword “flow” (required if flow problem).
<i>ja, jb, jc</i>	integer	See description under keyword “cond” (page 32).
<i>skd</i>	real	Depending on problem type: Heat- and/or mass-source strength (kg/s), heat only (MJ/s); Specified source air pressure (MPa); or Pressure above which outflow occurs (MPa).
<i>eflow</i>	real	Depending on problem type: Enthalpy of fluid injected (MJ/kg); Temperature (°C); Source liquid saturation (kg/s); or Source air pressure (MPa).
<i>aiped</i>	real	Impedance parameter.
A blank line to signal end of flow input.		

## DRAFT 4/97

**Table 3. Main input data file (continued)**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>	character*4	Keyword “ <b>flo2</b> ” (optional).
<i>ja, jb, jc, jd</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>skd</i>	real	Same as above for keyword “ <b>flow</b> ” (page 34).
<i>eflow</i>	real	Same as above for keyword “ <b>flow</b> ”.
<i>aiped</i>	real	Same as above for keyword “ <b>flow</b> ”.
A blank line to signal end of flo2 input.		
<i>macro</i>	character*4	Keyword “ <b>flox</b> ” (optional).
<i>nflx</i>	integer	Number of internode fluxes to be calculated.
<i>iflx1</i>	integer	First node to be used in flux calculation.
<i>iflx2</i>	integer	Second node to be used in flux calculation.
<i>x1</i>	real	X-, y-, and z-coordinates of the first node to be used in flux calculation.
<i>y1</i>	real	Used only for those nodes where IFLX1 < 0.
<i>z1</i>	real	
<i>x2</i>	real	X-, y-, and z-coordinates of the second node to be used in flux calculation. Used only for those nodes where IFLX2 < 0.
<i>y2</i>	real	
<i>z2</i>	real	
<i>macro</i>	character*4	Keyword “ <b>hflux</b> ” (optional).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>qflux</i>	real	If <i>qflxm</i> = 0, then <i>qflux</i> is the heat flux (MW). If <i>qflxm</i> ≠ 0, then <i>qflux</i> is a temperature (°C) and the heat flux is calculated according to the formula: $Q_H = qflxm(TL - qflux)$ (MW).
<i>qflxm</i>	real	If <i>qflxm</i> ≠ 0, multiplier for heat-flux equation given in description (MW/°C).
A blank line to signal end of heat-flux input.		
<i>macro</i>	character*4	Keyword “ <b>ice</b> ” (optional).
<i>ice</i>	integer	Solution descriptor for ice solution.
<i>siin</i>	real	Default value for ice saturation.
<i>tmelt</i>	real	Freezing temperature of water (°C).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>sii</i>	real	Ice saturation.
A blank line to signal end of ice-saturation input.		
<i>macro</i>	character*4	Keyword “ <b>init</b> ” (required if “ <b>pres</b> ” not used).
<i>pein</i>	real	Initial value of pressure (MPa).
<i>tin</i>	real	Initial value of temperature (°C).
<i>tin1</i>	real	Temperature-gradient linear formula parameter (°C).
<i>grad1</i>	real	Temperature-gradient linear formula parameter (°C/m).
<i>depth</i>	real	Temperature-gradient formula parameter (m).
<i>tin2</i>	real	Temperature-gradient quadratic formula parameter (°C).
<i>grad2</i>	real	Temperature-gradient quadratic formula parameter (°C/m).
<i>quad</i>	real	Temperature-gradient quadratic formula parameter (°C/m <sup>2</sup> ).

**Table 3. Main input data file (continued)**

Input variable	Format	Description
<i>macro</i>	character*4	Keyword “ <b>iter</b> ” (optional).
<i>g1</i>	real	Multiplier for the linear convergence region of the Newton-Raphson iteration.
<i>g2</i>	real	Multiplier for the quadratic convergence region of the Newton-Raphson iteration.
<i>g3</i>	real	Tolerance for the adaptive implicit method (multiplying factor for Newton-Raphson tolerance).
<i>tmch</i>	real	Machine tolerance. If satisfied by the residual norm, the Newton iteration is assumed to be complete.
<i>overf</i>	real	Over-relaxation factor for passive nodes in adaptive implicit method.
<i>irdof</i>	integer	Enables the reduced degree-of-freedom method.
<i>islord</i>	integer	Reordering parameter.
<i>iback</i>	integer	IRDOF parameter.
<i>icoupl</i>	integer	Number of SOR iterations used in reduced degree-of-freedom methods.
<i>rnmax</i>	real	Maximum running time for problem before the solution is stopped (cpu minutes).
<i>macro</i>	character*4	Keyword “ <b>itup</b> ” (optional).
<i>iad_up</i>	integer	Number of iterations after which the upwind directions are held constant.
<i>macro</i>	character*4	Keyword “ <b>iupk</b> ” (optional; no input is associated with this macro).
<i>macro</i>	character*4	Keyword “ <b>ivfc</b> ” (not supported this version).
<i>macro</i>	character*4	Keyword “ <b>mdnode</b> ” (optional).
<i>num_md</i>	integer	Number of new connections to be entered.
<i>max_con</i>	integer	Maximum number of new connections to a given node.
For each new connection ( <i>num_md</i> ):		
<i>node</i>	integer	Node to which new connection is established.
<i>ipar</i>	integer	Not used at present. Its value is ignored but the number entered must be an integer.
<i>npar</i>	integer	The new connected node. If <i>npar</i> = <i>node</i> , no new connection is established.
<i>macro</i>	character*4	Keyword “ <b>ngas</b> ” (optional).
<i>ico2d</i>	integer	Solution descriptor for noncondensable gas transport.
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>pco2</i>	real	Initial partial pressure of noncondensable gas. If <i>pco2</i> < 0, then ABS ( <i>pco2</i> ) is interpreted as a temperature and the partial pressure of the noncondensable gas is calculated.
A blank line to signal end of partial-pressure input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>cpnk</i>	real	If <i>cpnk</i> < 0, then ABS ( <i>cpnk</i> ) is the specified noncondensable pressure and will be held at that value. If <i>cpnk</i> > 0, then <i>cpnk</i> is the specified relative humidity and the saturation is calculated using the vapor-pressure lowering formula and the capillary-pressure formula.
A blank line to signal end of saturation input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>qcd</i>	real	Specified air flow rate (kg/s).
A blank line to signal end of air-flow-rate input.		

## DRAFT 4/97

**Table 3. Main input data file (continued)**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>	character*4	Keyword “ <b>nod2</b> ” (required if macro node is not used).
<i>m</i>	integer	Number of nodes for which information will be printed to output files.
<i>m2</i>	integer	Number of nodes for short list (terminal printout).
<i>mn(i)</i>	integer	<i>m</i> node numbers for which information will be printed on the output file ( <i>i</i> = 1 to <i>m</i> ). If a <i>mn(i)</i> < 0, then coordinates are used to define that node.
<i>x, y, z</i>	real	Coordinates of node for which information will be printed. One line for each <i>mn</i> < 0. The code finds the node closest to the coordinate given.
<i>mni(i)</i>	integer	<i>m2</i> node numbers for which information will be printed on the output file ( <i>i</i> = 1 to <i>m2</i> ). If a <i>mni(i)</i> < 0, then coordinates are used to define that node.
<i>x, y, z</i>	real	Coordinates of node for which information will be printed. One line for each <i>mni</i> < 0. The code finds the node closest to the coordinate given.
<i>macro</i>	character*4	Keyword “ <b>node</b> ” (required if macro nod2 is not used).
<i>m</i>	integer	Number of nodes for which information will be printed to output files.
<i>mn(i)</i>	integer	<i>m</i> node numbers for which information will be printed on the output file ( <i>i</i> = 1 to <i>m</i> ). If a <i>mn(i)</i> < 0, then coordinates are used to define that node.
<i>x, y, z</i>	real	Coordinates of node for which information will be printed. One line for each <i>mn</i> < 0. The code finds the node closest to the coordinate given.
or		
<i>keyword</i>	character*5	Keyword “ <b>block</b> ” to invoke node specification by ja, jb, jc format.
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
A blank line to signal end of node-block input.		
<i>macro</i>	character*4	Keyword “ <b>num</b> ” (not implemented).
<i>macro</i>	character*4	Keyword “ <b>perm</b> ” (required).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>pndx</i>	real	Permeability in the x-direction ( $\text{m}^2$ ).
<i>pnxd</i>	real	Permeability in the y-direction ( $\text{m}^2$ ).
<i>pnzd</i>	real	Permeability in the z-direction ( $\text{m}^2$ ).
A blank line to signal end of permeability input.		
<i>macro</i>	character*4	Keyword “ <b>ppor</b> ” (optional).
<i>iporos</i>	integer	Porosity/permeability type.
<i>r1</i>	real	Parameter used in the linear and Gangi models. For the linear model, pore volume compressibility, $c_r$ ( $\text{MPa}^{-1}$ ). For the Gangi model, coefficient of thermal expansion, $\alpha$ ( $^\circ\text{C}^{-1}$ ).
<i>r2</i>	real	Parameter used in the linear and Gangi models. For the linear model, compressibility of the matrix grain, $c_g$ ( $\text{MPa}^{-1}$ ). For the Gangi model, Young's modulus, $E$ (MPa).
<i>r3</i>	real	Parameter used in the Gangi model, initial stress, $\sigma$ (MPa).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>r4</i>	real	Variable parameter used in Gangi model, exponent <i>m</i> .
<i>r5</i>	real	Variable parameter used in Gangi model, pressure $P_0$ (MPa).
A blank line to signal end of Gangi-model parameter input.		

**DRAFT 4/97**

**Table 3. Main input data file (continued)**

Input variable	Format	Description
<i>macro</i>	character*4	Keyword “ <b>pres</b> ” (required if macro init not used).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>phrd</i>	real	Initial pressure (MPa).
<i>tind</i>	real	Initial temperature (°C) or initial saturation depending on thermodynamic region.
<i>ieosd</i>	integer	Thermodynamic-region parameter.
A blank line to signal end of initial-value input.		
<i>macro</i>	character*4	Keyword “ <b>ptrk</b> ” (optional).
<i>npart</i>	integer	Number of particles in the simulation.
<i>rseed</i>	integer	6-digit integer random-number seed.
<i>daycs</i>	real	Time for which the particle-tracking solution is enabled (days).
<i>daycf</i>	real	Time for which the particle-tracking solution is disabled (days).
<i>dayhf</i>	real	Time for which the flow solution is disabled (days).
<i>dayhs</i>	real	Time for which the flow solution is enabled (days).
<i>trak_type</i>	integer	Flag to denote the fluid phase of the particles.
<i>half_life</i>	real	Half-life for irreversible first-order decay reaction (s).
<i>pout</i>	integer	Flag to specify the concentration output.
<i>prnt_rst</i>	integer	Flag to specify whether particle information is written to the “.fin” file.
<i>transflag</i>	integer	Flag to specify which transport mechanisms apply.
<i>kd</i>	real	Sorption coefficient (linear, reversible, equilibrium sorption) (kg-fluid/kg-rock).
<i>tclx</i>	real	Dispersivity in the x-direction (m).
<i>tcly</i>	real	Dispersivity in the y-direction (m).
<i>tclz</i>	real	Dispersivity in the z-direction (m).
<i>diffmat</i>	real	Molecular diffusion coefficient in the rock matrix (m <sup>2</sup> /s).
<i>rd_frac</i>	real	Retardation factor within the primary porosity (fractures) for a matrix-diffusion particle-tracking simulation.
<i>matrix_por</i>	real	Porosity of the rock matrix.
<i>fspacing</i>	real	Mean fracture spacing (m).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>itrc</i>	integer	Model number for parameters defined above.
A blank line to signal end of model-number input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>pcnsk</i>	real	Particle-injection parameter.
<i>t1sk</i>	real	Time (days) when particle injection begins.
<i>t2sk</i>	real	Time (days) when particle injection ends.
A blank line to signal end of particle-injection parameter input.		
<i>macro</i>	character*4	Keyword “ <b>renm</b> ” (optional).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>igd</i>	integer	New node number for given node.
A blank line to signal end of renumbering input.		
<i>macro</i>	character*4	Keyword “ <b>rfix</b> ” (not implemented in this version of FEHM).

## DRAFT 4/97

**Table 3. Main input data file (continued)**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>	character*4	Keyword “rlp” (optional).
<i>irlp(i)</i>	integer	Relative-permeability model type.
<i>rp1</i>	real	Parameter used in all models. For models 1 and 2, irreducible liquid saturation. For models 3 and 4, residual liquid saturation.
<i>rp2</i>	real	Parameter used in all models. For models 1 and 2, irreducible vapor saturation. For models 3 and 4, maximum liquid saturation.
<i>rp3</i>	real	Parameter used in models 1, 3, and 4. For model 1, maximum liquid saturation. For models 3 and 4, $\alpha_G$ parameter (1/m) (note: some data are given in (1/Pa); convert using pressure = $\rho g \Delta h$ ).
<i>rp4</i>	real	Parameter used in models 1, 3, and 4. For model 1, maximum vapor saturation. For models 3 and 4, parameter n.
<i>rp5</i>	real	Parameter used in models 3 and 4. Multiple of cutoff capillary pressure assigned as maximum capillary pressure.
<i>rp6</i>	real	Parameter used in models 3 and 4. Cutoff saturation <i>rp1</i> .
<i>rp7</i>	real	Parameter used in model 4. Residual liquid saturation for fracture.
<i>rp8</i>	real	Parameter used in model 4. Maximum liquid saturation for fracture.
<i>rp9</i>	real	Parameter used in model 4. $\alpha_G$ for fractures.
<i>rp10</i>	real	Parameter used in model 4. n for fractures.
<i>rp11</i>	real	Parameter used in model 4. Same as <i>rp5</i> except for fractures.
<i>rp12</i>	real	Parameter used in model 4. Same as <i>rp6</i> except for fractures.
<i>rp13</i>	real	Parameter used in model 4. Fracture intrinsic permeability ( $m^2$ ).
<i>rp14</i>	real	Parameter used in model 4. Matrix intrinsic permeability ( $m^2$ ).
<i>rp15</i>	real	Parameter used in model 4. Fracture porosity.
<i>ja, jb, jc</i>	integer	See description under keyword “cond” (page 32).
<i>i</i>	integer	Number referring to the sequence of models read.
A blank line to signal end of relative-permeability model input.		
<i>rlpfile</i>	character*100	Name of the optional rlp data file used when <i>irlp(i) = 5</i> .
<i>macro</i>	character*4	Keyword “rock” (required).
<i>ja, jb, jc</i>	integer	See description under keyword “cond” (page 32).
<i>denrd</i>	real	Rock density ( $kg/m^3$ ).
<i>cprd</i>	real	Rock specific heat ( $\frac{MJ}{kg \cdot K}$ ).
<i>psd</i>	real	Porosity.
A blank line to signal end of rock input.		
<i>macro</i>	character*4	Keyword “rxn” (optional).
<i>key_group</i>	character	Key word to specify that species are to be placed into groups that are solved simultaneously.
<i>ngrps</i>	integer	Number of groups of species.
<i>group</i>	integer	<i>nspecies</i> values are entered for each line of input, and <i>ngrps</i> lines of input are required, one for each group.
<i>nrxns</i>	integer	Number of chemical reactions.
<i>rxn_interval</i>	real	Time between each iteration.
<i>key_rxn</i>	character*4	Denotes the type of reaction. The first letter of the keyword is all that is required.

**DRAFT 4/97**

**Table 3. Main input data file (continued)**

Input variable	Format	Description
<i>equil_model</i>	integer	Flag denoting which model is to be used for defining the temperature dependence of the equilibrium constant.
<i>equil_const25</i>	real	The term $A_{rxn}$ in equilibrium constant-temperature-dependence model.
<i>enthalpy25</i>	real	The term $\Delta H_H$ in equilibrium constant-temperature-dependence model.
<i>gamma_check</i>	real	Equilibrium-tolerance parameter $\gamma_{tol}$ .
<i>rate_factor</i>	real	Parameter for scaling the rate constants.
<i>round_tol</i>	real	Cut-off parameter for forward reaction rate.
or		
<i>key_rxn</i>	character*4	Same as above.
<i>equil_model</i>	integer	Same as above.
<i>awwa(1)</i>	real	The term $A_{rxn,1}$ in equilibrium constant-temperature-dependence model.
<i>awwa(2)</i>	real	The term $A_{rxn,2}$ in equilibrium constant-temperature-dependence model.
<i>awwa(3)</i>	real	The term $A_{rxn,3}$ in equilibrium constant-temperature-dependence model.
<i>awwa(4)</i>	real	The term $A_{rxn,4}$ in equilibrium constant-temperature-dependence model.
<i>awwa(5)</i>	real	The term $A_{rxn,5}$ in equilibrium constant-temperature-dependence model.
<i>gamma_check</i>	real	Same as above.
<i>rate_factor</i>	real	Same as above.
<i>round_tol</i>	real	Same as above.
or		
<i>key_rxn</i>	character*4	Same as above.
<i>ar_for</i>	real	Pre-exponential factor of the forward reaction.
<i>ea_for</i>	real	Activation energy of the forward reaction (kJ/mol).
<i>ar_rev</i>	real	Pre-exponential factor of the reverse reaction.
<i>ea_rev</i>	real	Activation energy of the reverse reaction (kJ/mol).
<i>stoic</i>	real	The stoichiometric coefficient.
<i>rate_power</i>	real	The exponent in the rate law.
<i>fl_mult</i>	real	Parameter signifying whether this particular reaction occurs for this solute in the fluid phase.
<i>sb_mult</i>	real	Parameter signifying whether this particular reaction occurs for this solute in the sorbed phase.
<i>h_mult</i>		For Henry's Law species, parameter signifying whether this particular reaction occurs for the liquid-borne or vapor-borne portion of the species.
<i>macro</i>	character*4	Keyword " <b>sol</b> " (required).
<i>ntt</i>	integer	Parameter that defines the type of solution required.
<i>intg</i>	integer	Parameter that defines element integration type.
<i>macro</i>	character*4	Keyword " <b>solv</b> " (not implemented).
<i>macro</i>	character*4	Keyword " <b>stea</b> " (optional; no input is associated with this macro).
<i>macro</i>	character*4	Keyword " <b>strs</b> " (not implemented in this version of FEHM).
<i>macro</i>	character*4	Keyword " <b>text</b> " (optional).
<i>wdd1</i>	character*80	Line of text. A maximum of 80 characters per line are entered.
A blank line to signal end of text input.		

## DRAFT 4/97

**Table 3. Main input data file (continued)**

<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>	character*4	Keyword “ <b>thic</b> ” (optional).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>thic</i>	real	Thickness of the model domain in the third dimension (m).
A blank line to signal end of thickness input.		
<i>macro</i>	character*4	Keyword “ <b>time</b> ” (required).
<i>day</i>	real	Initial time-step size (days).
<i>tims</i>	real	Final simulation time (days).
<i>nstep</i>	integer	Maximum number of time steps allowed.
<i>ii</i>	integer	Print-out interval for nodal information (pressure, enthalpy, etc.) as set up under control-statement node (i.e., number of time steps).
<i>year</i>	integer	Year that simulation starts.
<i>month</i>	integer	Month that simulation starts.
<i>dit1</i>	real	Time (days) for time-step change.
<i>dit2</i>	real	New time-step size (days).
<i>dit3</i>	real	Implicitness factor for new time step.
<i>itc</i>	integer	New print-out interval.
A blank line to signal end of time-step change input.		
<i>macro</i>	character*4	Keyword “ <b>trac</b> ” (optional).
<i>user_macro</i>	character*5	Key word for invoking a solute-transport user subroutine.
<i>ano</i>	real	Initial solute concentration.
<i>awc</i>	real	Implicitness factor for solute solution.
<i>epc</i>	real	Equation tolerance for solute solution.
<i>upwgta</i>	real	Upstream weighting term for the solute solution.
<i>daycs</i>	real	Time for which the solute solution is enabled (days).
<i>daycf</i>	real	Time for which the solute solution is disabled (days).
<i>dayhf</i>	real	Time for which the flow solution is disabled (days).
<i>dayhs</i>	real	Time for which the flow solution is enabled (days).
<i>iaccmx</i>	integer	Maximum number of iterations allowed in solute solution if time-step multiplier is enabled.
<i>daycm</i>	real	Time-step multiplier for solute solution.
<i>daycmm</i>	real	Minimum time step for solute solution (days).
<i>daycmx</i>	real	Maximum time step for solute solution (days).
<i>nspecies</i>	integer	Number of solutes simulated.
<i>input_msg</i>	character*4	Keyword ‘ <i>ldsp</i> ’ specifying longitudinal/transverse dispersion is desired.
There are two options for dispersivity and sorption parameters.		
<i>icns</i>	integer	Phase designation for the <i>i</i> th solute.
<i>iadsf</i>	integer	Adsorption model type for the <i>i</i> th species, <i>i</i> th region.
<i>a1adsf</i>	real	$\alpha_1$ parameter in adsorption model.
<i>a2adsf</i>	real	$\alpha_2$ parameter in adsorption model.
<i>betadf</i>	real	$\beta$ parameter in adsorption model.
<i>diffm</i>	real	Molecular diffusion coefficient ( $m^2/s$ ).
<i>tcx, tcy, tcz</i>	real	Dispersivity in the x-, y-, z-directions (m).
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>itrcd</i>	integer	Region number for sorption and dispersion parameters given above.
A blank line to signal end of region input.		

**DRAFT 4/97**

**Table 3. Main input data file (continued)**

Input variable	Format	Description
or		
<i>input_msg</i>	character*4	Keyword specifying that the same diffusion coefficient and dispersivities are to be used for all species of the same type (liquid and/or vapor): 'dspl' - liquid only, 'dspv' - vapor only, 'dspb' - both liquid and vapor.
<i>diffm</i>	real	Molecular diffusion coefficient ( $m^2/s$ ) for liquid or vapor only.
<i>tcx, tcy, tcz</i>	real	Dispersivity in the x-, y-, z-directions (m) for liquid or vapor only.
or		
<i>diffml</i>	real	Molecular diffusion coefficient ( $m^2/s$ ) for liquid.
<i>tcxl, tcyl, tczl</i>	real	Dispersivity in the x-, y-, z-directions (m) for liquid.
<i>diffmv</i>	real	Molecular diffusion coefficient ( $m^2/s$ ) for vapor.
<i>tcxv, tcyv, tczv</i>	real	Dispersivity in the x-, y-, z-directions (m) for vapor.
<i>ja, jb, jc</i>	integer	See description under keyword "cond" (page 32).
<i>itrcdsp</i>	integer	Region number for dispersion parameters given above.
A blank line to signal end of region input.		
<i>icns</i>	integer	Phase designation for the <i>i</i> th solute.
<i>iadsf</i>	integer	Adsorption model type for the <i>i</i> th species, <i>i</i> th region.
<i>a1adsf</i>	real	$\alpha_1$ parameter in adsorption model.
<i>a2adsf</i>	real	$\alpha_2$ parameter in adsorption model.
<i>betadf</i>	real	$\beta$ parameter in adsorption model.
<i>ja, jb, jc</i>	integer	See description under keyword "cond" (page 32).
<i>itrcd</i>	integer	Region number for sorption parameters given above.
A blank line to signal end of region input.		
The following data apply to either of the sorption/dispersivity input options used above.		
<i>henry_model</i>	integer	Flag denoting which model is to be used for defining the temperature dependence of the Henry's Law constant.
<i>hawwa(1)</i>	real	Term in Henry's Law temperature-dependence model: model 1 - $A_H$ , model 2 - $A_{H,1}$
<i>hawwa(2)</i>	real	Term in Henry's Law temperature-dependence model: model 1 - $\Delta H_H$ , model 2 - $A_{H,2}$
<i>hawwa(3)</i>	real	Term in Henry's Law temperature-dependence model 2 - $A_{H,3}$ .
<i>hawwa(4)</i>	real	Term in Henry's Law temperature-dependence model 2 - $A_{H,4}$ .
<i>hawwa(5)</i>	real	Term in Henry's Law temperature-dependence model 2 - $A_{H,5}$ .
<i>ja, jb, jc</i>	integer	See description under keyword "cond" (page 32).
<i>anqo</i>	real	Initial concentration of tracer, which will supersede the value given by <i>ano</i> .
A blank line to signal end of concentration input.		
<i>ja, jb, jc</i>	integer	See description under keyword "cond" (page 32).
<i>cnsk</i>	real	Injection concentration at inlet node (moles per kg liquid or vapor).
<i>t1sk</i>	real	Time (days) when tracer injection begins.
<i>t2sk</i>	real	Time (days) when tracer injection ends.
A blank line to signal end of tracer injection input.		
<i>macro</i>	character*4	Keyword "user".
<i>kk</i>	integer	Integer number passed to subroutine user for user-defined input parameters.

## DRAFT 4/97

**Table 3. Main input data file (continued)**

Input variable	Format	Description
<i>macro</i>	character*4	Keyword “ <b>vcon</b> ”.
<i>ivcon(i)</i>	integer	Model type for <i>i</i> th conductivity model: linear variation of thermal conductivity with temperature or square-root variation of thermal conductivity with liquid saturation.
<i>vc1f(i)</i>	real	Reference temperature ( $^{\circ}\text{C}$ ) or conductivity ( $\frac{\text{W}}{\text{m} \cdot \text{K}}$ ) at liquid saturation = 1, depending on model.
<i>vc2f(i)</i>	real	Reference conductivity ( $\frac{\text{W}}{\text{m} \cdot \text{K}}$ ) or conductivity ( $\frac{\text{W}}{\text{m} \cdot \text{K}}$ ) at liquid saturation = 0, depending on model.
<i>vc3f(i)</i>	real	Change in conductivity with respect to temperature or not used, depending on model.
A blank line to signal end of conductivity-model input.		
<i>ja, jb, jc</i>	integer	See description under keyword “ <b>cond</b> ” (page 32).
<i>ivcnd</i>	integer	Number referring to the sequence of models read.
A blank line to signal end of conductivity-model desingator input.		
<i>macro</i>	character*4	Keyword “ <b>velo</b> ”. The input is identical to “ <b>flxo</b> ” except that velocities instead of fluxes are calculated (see page 35).
<i>macro</i>	character*4	Keyword “ <b>wlbr</b> ” (not supported this version).
<i>macro</i>	character*4	Keyword “ <b>zone</b> ” (optional). See Section 4.1.4.2.4 on page 44.
<i>macro</i>	character*4	Keyword “ <b>stop</b> ”.

**4.1.4.2.3 Geometry data file.** The geometry data file contains the mesh element and coordinate data (Table 4). This information can either be part of the main input file or a separate file.

**Table 4. Geometry data file**

Input variable	Format	Description
<i>macro</i>	character*4	Keyword “ <b>coor</b> ”.
<i>n</i>	integer	Number of nodes in the grid.
For each node:		
<i>mb</i>	integer	Node number. If <i>mb</i> < 0, then the difference between the absolute value of <i>mb</i> and the previously read absolute value of <i>mb</i> is used to generate intermediate values by interpolation.
<i>cord(mb,1)</i>	real	X-coordinate of node <i>mb</i> (m).
<i>cord(mb,2)</i>	real	Y-coordinate of node <i>mb</i> (m).
<i>cord(mb,3)</i>	real	Z-coordinate of node <i>mb</i> (m).
A blank line to signal end of node input.		
<i>macro</i>	character*4	Keyword “ <b>elem</b> ”.
<i>ns</i>	integer	Number of nodes per element.
<i>nei</i>	integer	Number of elements.

## DRAFT 4/97

<b>Table 4. Geometry data file (continued)</b>		
<b>Input variable</b>	<b>Format</b>	<b>Description</b>
For each element:		
<i>mb</i>	integer	Element number. If <i>mb</i> < 0, then the difference between the absolute value of <i>mb</i> and the previous absolute value of <i>mb</i> is used to generate intermediate values by interpolation in the code.
<i>nelm (k)</i>	integer	<i>ns</i> node numbers for the nodes in element <i>mb</i> ( <i>k</i> = ( <i>mb</i> - 1)* <i>ns</i> + <i>i</i> , <i>i</i> = 1 to <i>ns</i> ).
A blank line to signal end of element input.		
<i>macro</i>	character*4	Keyword “ <b>stop</b> ”.

**4.1.4.2.4 Zone data file.** The zone data file contains the zone information, which can either be part of the main input file or a separate file (Table 5).

<b>Table 5. Zone data file</b>		
<b>Input variable</b>	<b>Format</b>	<b>Description</b>
<i>macro</i>		
<i>macro</i>	character*4	Keyword “ <b>zone</b> ”.
For each geometric zone:		
<i>izone</i>	integer	Zone identification number.
Followed by:		
<i>xz(i)</i>	real	X-coordinates defining zone <i>izone</i> .
<i>yz(i)</i>	real	Y-coordinates defining zone <i>izone</i> .
<i>zz(i)</i>	real	Z-coordinates defining zone <i>izone</i> . Note: <i>i</i> = 1 to 4 for 2-D and <i>zz</i> is not input, <i>i</i> = 1 to 8 for 3-D.
or		
<i>macro</i>	character*4	Keyword “ <b>list</b> ”.
<i>xg</i>	real	A list of x-, y-, z-coordinates, one set per line until a blank line is encountered ( <i>zg</i> is not input for 2D). The nodes corresponding to these coordinates make up <i>izone</i> .
<i>yg</i>	real	
<i>zg</i>	real	
or		
<i>macro</i>	character*4	Keyword “ <b>nnum</b> ”.
<i>nin</i>	integer	Specified number of nodes.
<i>ncord(i)</i>	integer	<i>nin</i> node numbers of the nodes to be included in <i>izone</i> ( <i>i</i> = 1 to <i>nin</i> ).
A blank line to signal end of zone input.		
<i>macro</i>	character*4	Keyword “ <b>stop</b> ”.

**4.1.4.2.5 Optional input files.** The optional input files contain data for specified macros using the macro formats described above in Section 4.1.4.2.2 and Section 4.1.4.2.4 of this chapter.

**4.1.4.2.6 Restart input file.** The restart input data file provides **fehm** initial values of pressure, temperature, saturation, and simulation or starting time. See Section 4.1.4.4.2 (Table 7) for variable names, format, and description. The code version, date, time, and problem title contained in the file are not used for input, but that information or two blank lines must be present for the file to be read correctly.

## DRAFT 4/97

**4.1.4.2.7 Coefficient storage file.** The coefficient storage input file contains finite-element coefficients calculated by the code during a previous run. See Section 4.1.4.4.7 (Table 9) of this chapter for variable names, format, and description. The code version, date, time, and problem title contained in the file are not used for input, but that information or two blank lines must be present for the file to be read correctly.

### 4.1.4.3 Prompted input

Input prompts are used only when the default control file (*fehmn.files*) is not present in the directory from which the code is being executed. Table 6 below lists the prompts and expected responses. If the initial prompt is answered with the name of an input control file (see Section 4.1.4.2.1 in this chapter above), no further prompts will be issued and the remainder of the I/O data will be read from the control file. If the I/O data is not accepted (final prompt answered “no”), the prompts will be repeated starting with request for the control file name. The terminal display that accompanies input prompting is shown in Section 4.1.4.5.1 of this chapter.

A carriage return, <cr>, refers to a return with no input that accepts the displayed default value. Responses to file name queries may be the name of a file (including path or subdirectories), <cr>, or either “na” or “not using” to indicate the file is not required by the user. Prompts that use the input file name response are denoted <input> or <root> (input file name without suffix).

**Table 6. Prompted input**

Input prompt	Format	Expected response
Enter name for ioctl -- default file name: not using [(name/na or not using), RETURN = DEFAULT]	character*100	Input control file name, <cr>, or either “na” or “not using”.
Enter name for inpt -- default file name: fehmn.dat [(name/na or not using), RETURN = DEFAULT]	character*100	Input file name or <cr> to use the default name.
Do you want all file names of the form <root of input file name>.* ? [(y/n), RETURN = y] *** Note: If “y” incoor and inzone will equal inpt ***	character*100	“y” or “n”.
Note: The next 11 filename prompts are issued only if the previous prompt was answered with “n”.		
Enter name for incoor -- default file name: <input> [(name/na or not using), RETURN = DEFAULT]	character*100	Geometry data file name, <cr>, or either “na” or “not using”.
Enter name for inzone -- default file name: <input> [(name/na or not using), RETURN = DEFAULT]	character*100	Zone data file name, <cr>, or either “na” or “not using”.
Enter name for iout -- default file name: <root>.out [(name/na or not using), RETURN = DEFAULT]	character*100	Output file name or <cr> to use the default name.
Enter name for iread -- default file name: <root>.ini [(name/na or not using), RETURN = DEFAULT]	character*100	Restart input file name, <cr>, or either “na” or “not using”.

**Table 6. Prompted input (continued)**

Input prompt	Format	Expected response
Enter name for isave -- default file name: <root>.fin [(name/na or not using), RETURN = DEFAULT]	character*100	Restart output file name, <cr>, or either "na" or "not using".
Enter name for ishis -- default file name: <root>.his [(name/na or not using), RETURN = DEFAULT]	character*100	History plot file name, <cr>, or either "na" or "not using".
Enter name for istrc -- default file name: <root>.trc [(name/na or not using), RETURN = DEFAULT]	character*100	Solute plot file name, <cr>, or either "na" or "not using".
Enter name for iscon -- default file name: <root>.con [(name/na or not using), RETURN = DEFAULT]	character*100	Contour plot file name, <cr>, or either "na" or "not using".
Enter name for iscon1 -- default file name: <root>.dp [(name/na or not using), RETURN = DEFAULT]	character*100	Contour plot file name for dual or dpdp, <cr>, or either "na" or "not using".
Enter name for issstor -- default file name: <root>.stor [(name/na or not using), RETURN = DEFAULT]	character*100	Coefficient storage file name, <cr>, or either "na" or "not using".
Enter name for ischk -- default file name: <root>.chk [(name/na or not using), RETURN = DEFAULT]	character*100	Input check output file name, <cr>, or either "na" or "not using".
tty output -- show all reference nodes, selected reference nodes, or none: [(all/some/none), RETURN = none]	character*4	"all", "some", or "none".
user subroutine number (provided to subroutine USER before every time step): [RETURN = none]	integer	Number of user subroutine, <cr>, or "0" for not using.
If data is OK enter yes to continue, no to restart terminal input, or stop to end program: [(yes/no/stop), RETURN = yes]	character*4	"yes", "no", or "stop".

#### 4.1.4.4 Output files

**4.1.4.4.1 Main output file.** The main output file records the code version, date, and time followed by the user-input problem title. A summary of the I/O files used, macro control statements read, and array storage follow. Timing, equation performance, nodal (for user-specified nodes), and global mass- and energy-balance information at user-selected time intervals is written in a narrative format. See Section 4.1.4.5.2 in this chapter for an example of the information output and narrative style. The file ends with a summary of simulation time, number of time steps in the problem, the number of iterations taken, and total cpu time. The default output file name is *fehmn.out*.

**4.1.4.4.2 Restart output file.** The restart output data file stores **fehm** final values of pressure, temperature, saturation, and simulation time (Table 7). The default restart output file name is *fehmn.fin*.

## DRAFT 4/97

**Table 7. Restart output file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
days	real*8	Simulation time (days)
wdd1(1:4)	character*4	Gas problem flag (ngas, h20, air)
wdd1(5:8)	character*4	Tracer problem flag (trac, ntra)
wdd1(9:12)	character*4	Stress problem flag (strs, nstr)
wdd1(13:16)	character*4	Double-porosity/double-permeability problem flag (dpdp, ndpd)
wdd1(17:20)	character*4	Dual-porosity problem flag (dual, ndua)
t(ncount)	real*8	Temperature (not present when air flag is set, isothermal air/water problem)
s(ncount)	real*8	Liquid saturation
phi(ncount)	real*8	Pressure
pci(ncount)	real*8	Gas pressure (only present if noncondensable gas problem, ngas flag is set).

Array dimension ncount = neq, the number of nodes in the problem; if dpdp flag is set, ncount = 2\*neq; if dual flag is set, ncount = 3\*neq

**4.1.4.4.3 Simulation history output file.** The history plot file records parameter values at selected nodes for each time step of the simulation (Table 8). The default history output file name is *fehmn.his*.

**Table 8. Simulation history output file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
N/A	character*4	Tracer problem flag ('trac' or blank)
N/A	character*4	Stress problem flag ('strs' or blank)
m	integer	Number of nodes for which data are output
mi	integer	Node number
cord(mi,*)	real*8	X-, y-, and z-coordinate of each node for which data are output (m sets)
N/A	character	Data headings (3 lines) as follows: headings node flow enthalpy(Mj/kg) flow(kg/s) temperature(deg C) total pressure (Mpa) capillary pressure(Mpa) saturation(kg/kg)
For each time step:		
days	real*8	Simulation time

**Table 8. Simulation history output file (continued)**

Output variable	Format	Description
followed by (for each specified output node):		
mi	integer	Node number
qh(mi)	real*8	Energy source (MJ/s)
sk(mi)	real*8	Source strength (kg/s)
t(mi)	real*8	Temperature (°C)
phi(mi)	real*8	Pressure (MPa)
pcp(mi)	real*8	Capillary pressure (MPa)
s(mi)	real*8	Saturation (dimensionless)

**4.1.4.4 Solute history output file.** The solute history plot file records solute parameter values at selected nodes for each time step of the simulation (Table 9). The default solute history output file name is *fehmn.trc*.

**Table 9. Solute history output file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
m	integer	Number of nodes for which data are output
mi	integer	Node number
cord(mi,*)	real*8	X-, y-, and z-coordinate of each node for which data are output (m sets)
nspeci	integer	Number of different species for tracer solution
For each time step:		
days	real*8	Simulation time
nsp	integer	Species number
followed by (for each specified output node):		
an(mi)	real*8	Species concentration (dimensionless)

**4.1.4.5 Contour plot output file.** The contour plot records parameter values for each node at selected time steps of the simulation (Table 10). The default contour plot output file name is *fehmn.con*.

**Table 10. Contour plot output file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
N/A	character*4	Tracer problem flag ('trac' or blank)
N/A	character*4	Stress problem flag ('strs' or blank)
neq	integer	Number of nodes for which data are output
cord(i,*)	real*8	X-, y-, and z-coordinate of each node for which data is output (neq sets)

## DRAFT 4/97

**Table 10. Contour plot output file (continued)**

Output variable	Format	Description
ns	integer	Number of nodes per element, total number of elements
nei	integer	
nelm(ns)	integer	Nodal connectivity information for each node of each element
px(neq)	real*8	X-, y-, z-permeability ( $\text{m}^2$ ) for each node
py(neq)	real*8	
pz(neq)	real*8	
thx(neq)	real*8	X-, y-, z-thermal conductivity ( $\frac{\text{W}}{\text{m} \cdot \text{K}}$ ) for each node
thy(neq)	real*8	
thz(neq)	real*8	
ps(neq)	real*8	Porosity, rock specific heat ( $\frac{\text{MJ}}{\text{kg} \cdot \text{K}}$ ), capillary pressure (MPa) for
cpr(neq)	real*8	
pcp(neq)	real*8	
idof	integer	Number of degrees of freedom per node for the current problem
igrav	integer	Direction of gravity in problem
grav	real*8	Value of gravity
nspci	integer	Number of species if tracer solution is present
For each specified time step:		
days	real*8	Simulation time
inj	real*8	Injection phase ( $\geq 0$ : liquid; $< 0$ : vapor)
If injection phase is liquid, for each node:		
N/A	real*8	Liquid transmissibility/density [dil(neq)/rolf(neq)]
rolf(neq)	real*8	Liquid density
N/A	real*8	Pressure - capillary pressure [ $\phi(\text{neq}) - \text{pcp}(\text{neq})$ ]
t(neq)	real*8	Temperature
If injection phase is vapor, for each node:		
N/A	real*8	Vapor transmissibility/density [div(neq)/rovf(neq)]
rovf(neq)	real*8	Vapor density
phi(neq)	real*8	Pressure
t(neq)	real*8	Temperature
If tracer solution is present, for each species, for each node:		
an(neq) or anv(neq)	real*8	Species concentration of liquid phase or vapor phase

**4.1.4.4.6 Dual-porosity or double-porosity/double-permeability contour plot output file.** The dual-porosity or double-porosity/double-permeability contour plot file records parameter values for each dpdp (neq + 1 to 2\*neq) or dual (neq + 1 to 3\*neq) node at selected time steps of the simulation (Table 11). This file contains the same information as the regular contour plot file (see Table 10 above). The default dual-porosity or double-porosity/double-permeability contour plot output file name is *fehmnn.dp*.

## DRAFT 4/97

**4.1.4.4.7 Coefficient storage file.** The coefficient storage output file contains finite-element coefficients calculated by the code. The default coefficient storage file name is *fehmn.stor*.

**Table 11. Coefficient storage file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
iwtotl	integer	Number of storage locations needed to store geometric input types
neq	integer	Number of nodes
ncont	integer	Number of positions for which information needs to be stored
sx1(neq)	real*8	Volume associated with each node
nelm(ncont)	integer	Nodal-connectivity information for each connection
istrw(ncont)	integer	Starting position for each connection
nelmdg(neq)	integer	Position of element in connectivity array for each node
sx(iwtotl)	real*8	Finite-element geometric coefficient for each storage location
sxs(iwtotl)	real*8	Finite-element geometric coefficient for each storage location for stress module if enabled

**4.1.4.4.8 Input check output file.** The input check output file contains a summary of input information that may be of use in debugging or memory management of the code. The positions of maximum and minimum values of input parameters and derived quantities are given. Also provided is an analysis of array storage requirements. The default input check output file name is *fehmn.chk*.

**4.1.4.4.9 Error output file.** The error output file contains any error or warning messages issued by the code during a run (Table 12). The default error output file name is *fehmn.err*.

**Table 12. Error output file**

Output variable	Format	Description
verno	character*20	Code version
jdate	character*11	Date
jtime	character*8	Time
wdd	character*80	Problem title
N/A	character	Warning and/or error messages issued during code execution (see Section 4.1.4.6 for a summary of possible messages)

## DRAFT 4/97

**4.1.4.4.10 AVS log output file.** The AVS log output file provides a record of AVS data output files names and corresponding simulation time (Table 13).

**Table 13. AVS log output file**

Output variable	Format	Description
verno	character*2	"# "
jdate	character*20	Code version
jtime	character*11	Date
jtime	character*8	Time
N/A	character*80	Heading :# LOG AVS OUTPUT
wdd	character*2	"# "
wdd	character*80	Problem title
Blank line followed by (for each AVS data file written):		
root	character*95	Filename root
icall	integer	File number (appended to root)
days	real*8	Simulation time (days)

**4.1.4.4.11 AVS header output files.** The AVS ASCII header file contains information about the AVS output files and one line of header data as follows (Table 14).

**Table 14. AVS ASCII header output files**

Output variable	Format	Description
verno	character*2	"# "
jdate	character*20	Code version
N/A	character*11	Date
N/A	character*80	Heading :# AVS UNSTRUCTURED CELL DATA (UCD) FROM FEHM
wdd	character*2	"# "
wdd	character*80	Problem title
N/A	character	Text as follows: <pre># **** # To prepare files for input to avs one must # concatenate header/geometry/node_value files. # For example, if your FEHM input file was fe.dat, # headers are fe10001_sca_head fe10001_vec_head, ..., # mesh geometry will be in fe10001_geo, # field output will be in fe10001_sca_node, #           fe10001_vec_node, fe10001_con_dual_node # # A UCD input file can be produced using # cat fe10001_sca_head fe10001_geo fe10001_sca_node &gt; #       fe10001_sca_node.inp # # The UNIX for each command is useful for processing # multiple files. Also use the shell script fehm2avs # to perform automatic processing of all output. # ****</pre>

**DRAFT 4/97**

**Table 14. AVS ASCII header output files (continued)**

Output variable	Format	Description
neq	integer	Number of nodes
nei	integer	Number of cells (elements)
num_ndata	integer	Number of data components for the nodes
num_cdata	integer	Number of data components for the cells (0)
num_mdata	integer	Number of data components for the model (0)

The AVS binary header file contains only the header data as follows in 21 bytes (Table 15).

**Table 15. AVS binary header output files**

Output variable	Format	Description
	1 byte int	The number "7" indicating binary file
num_nodes	4 byte int	Number of nodes
num_cells	4 byte int	Number of cells (elements)
num_node_data	4 byte int	Number of data components for the nodes
num_cell_data	4 byte int	Number of data components for the cells (0)
num_model_data	4 byte int	Number of data components for the model (0)

**4.1.4.4.12 AVS geometry output file.** The AVS geometry file contains node, coordinate, cell, and material data. The ASCII file contains the data as follows (Table 16).

**Table 16. AVS ASCII geometry output file**

Output variable	Format	Description
i	integer	Node id
x, y, z	real*8	X-, y-, z-coordinate for each node
i	integer	Cell id
ipropelm	integer	Material id
char_type	character*5	Cell type
nelm	integer	List of cell vertices for each cell

The binary (unformatted) file contains the data as follows (Table 17).

**Table 17. AVS binary geometry output file**

Output variable	Format	Description
num_nlist	4 byte int	Number of nodes in connection list
i	4 byte int	Cell id
ipropm	4 byte int	Material id
ns	4 byte int	Num nodes
elem_type	4 byte int	Cell type - cell information for each cell
np	num_nlist 4 byte ints	Node list - cell topology (connectivity list)
x	num_nodes 4 byte floats	X-coordinates
y	num_nodes 4 byte floats	Y-coordinates
z	num_nodes 4 byte floats	Z-coordinates

## DRAFT 4/97

**4.1.4.4.13 AVS data output files.** The ASCII node data files contain the number of data components and size, a label/unit description for each data component, and the associated data for each node as follows (Table 18).

<b>Table 18. AVS ASCII data output files</b>		
<b>Output variable</b>	<b>Format</b>	<b>Description</b>
N/A	integer	Number of data components and size
title	character*80	Variable label and output units for each data component
Material properties data output for each node.		
i	integer	Node
px, pny, pnz	real*8	Permeability in x-, y-, and z-direction ( $\text{m}^2$ )
thx, thy, thz	real*8	Thermal conductivity in x-, y-, and z-direction ( $\frac{\text{W}}{\text{m} \cdot \text{K}}$ )
ps	real*8	Porosity
cpr	real*8	Rock specific heat ( $\frac{\text{MJ}}{\text{kg} \cdot \text{K}}$ )
pcp	real*8	Capillary pressure (MPa)
irlp	real*8	Relative-permeability model
icap	real*8	Capillary-pressure model
Scalar parameters that may be output for each node (up to 4).		
i	integer	Node
s	real	Saturation
t	real	Temperature ( $^{\circ}\text{C}$ )
phi	real	Liquid pressure (MPa)
phi - pcp	real	Vapor pressure (MPa)
Vector parameters that may be output for each node.		
pnxl, pnyl, pnzl	real	Liquid velocity (m/s), x-, y-, and z-component
pnxv, pnyv, pnzv	real	Vapor velocity (m/s), x-, y-, and z-component
Solute concentrations that may be output for each node (up to 20).		
an	real	Tracer concentration

The binary (unformatted) AVS data files use the following format for the data types described above (Table 19).

<b>Table 19. AVS binary data output files</b>		
<b>Output variable</b>	<b>Format</b>	<b>Description</b>
See previous table	1024 byte string	Node data labels
	1024 byte string	Node data units
	4 byte int	Number of node components
	4 byte floats	Node-component list
	4 byte floats	Minimums for node data
	4 byte floats	Maximums for node data
	4 byte floats	Data blocks with values for each node

## DRAFT 4/97

### 4.1.4.5 Displays and reports

4.1.4.5.1 **I/O prompt display.** The following file description display appears on the screen when **fehm** is executed and I/O prompts are invoked (see Section 4.1.4.3 in this chapter above).

```
version FEHM 01.00 [machine] 96/05/29 15:34:29
```

\*\*\*\* Default names for I/O files \*\*\*\*

```
control file      : fehmn.files
input file       : filen.*
geometry data file: filen.*
zone data file  : filen.*
output file      : filen.out
read file (if it exists): filen.ini
write file (if it exists): filen.fin
history plot file : filen.his
tracer history plot file: filen.trc
contour plot file: filen.con
dual or dpdp contour plot file: filen.dp
stiffness matrix data read/write file: filen.stor
input check file : filen.chk
```

\*\*\*\* where \*\*\*\*

“filen.\*” may be 100 characters maximum. If a name is not entered when prompted for, a default file name is used. “fehmn.dat” is the default used for the input file name.

\*\*\*\* note \*\*\*\*

A save file and input check file are always written. If you do not provide a name for these files, the following defaults will be used: fehmn.fin, fehmn.chk.

After terminal I/O has been completed, the following display reporting what was input (user-input responses shown in < >) is shown prior to the prompt asking if the I/O data are OK.

```
Not using tty output <none> or First reference output node will be written to tty <some> or All reference output nodes will be written to tty <all>
```

File purpose - Variable - Unit number - File name

```
control      - ioctl - 0 - not using
input        - inpt- 11 - <input file name>
geometry     - incoor- 11 - <geometry data file name>
```

## DRAFT 4/97

zone	- inzone- 11 - <zone data file name>
output	- iout- 14 - <output data file name>
initial state	- iread- 0 - <restart input file name>
final state	- isave- 16 - <restart output file name>
time history	- ishis- 17 - <history plot file name>
time his.(tr)	- istrc- 18 - <solute history plot file name>
contour plot	- iscon- 19 - <contour plot file name>
con plot (dp)	- iscon1- 20 - <dp contour plot file name>
fe coef stor	- isstor- 21 - <coefficient storage file name>
input check	- ischk- 22 - <input check output file name>

Value provided to subroutine user: <user subroutine number>

**4.1.4.5.2 Terminal run time display.** If terminal output is invoked (tty\_flag “all” or “some”), a summary of the I/O files used (as shown above in Section 4.1.4.5.1 of this chapter) is displayed. This information is followed by the maximum number of nodes in the problem, the problem title, macro control statements read, and array storage parameters as shown in the example on the next page. The output marked with a “bar” is displayed only when the “all” flag is used.

DRAFT 4/97

n0 = 9

## \*\*\*\*\* 2-D Heat Conduction Model \*\*\*\*\*

pressures and temperatures set by gradients

storage needed for ncon43 available43  
storage needed for nop43 available46  
storage needed for a matrix33 available33  
storage needed for b matrix33 available46  
storage needed for gmres81 available81  
storage available for b matrix resized to 33<<<<<

time for reading input, forming coefficients 0.667E-01

## \*\*\*\* analysis of input data on file fehmn.chk

\* \* \* \*

volumes and fe coefficients checked

storage for fe coefficients 12 allocated 12

Information that is displayed at each time step if either `tty_flag` is used is illustrated below. This information includes timing, equation performance, and mass and energy balances. The output marked with a “bar” is displayed at selected time steps for the selected output nodes only when the “all” flag is used. Additional parameters may be displayed depending on the type of problem (i.e., transport or double porosity) being executed.

## DRAFT 4/97

Time Step 1

### Timing Information

Years	Days	Step Size (Days)
0.136893E-04	0.500000E-02	0.500000E-02
Cpu Sec for Time Step = 0.3932E-02 Current Total = 0.1184E-01		

### Equation Performance

Number of N-R Iterations: 1  
 Avg # of Linear Equation Solver Iterations: 3.0  
 Number of Active Nodes: 9.  
 Total Number of Newton-Raphson Iterations: 1  
 Largest Residuals  
 EQ1 R= 0.1660E-07 node= 5 x=0.2500 y=0.2500 z= 1.000

Node	Equation 1 Residual	Equation 2 Residual
5	0.165983E-07	0.135450E+01
7	0.111444E-07	0.185894E-01

### Nodal Information (Water)

Node	p(MPa)	e(MJ)	I sat	temp(c)	source/sink (kg/s)	source/sink (MJ/s)
5	10.000	0.00	0.000	198.645	0.	0.
7	10.000	0.00	0.000	199.981	0.	0.
.						
.						
.						

Time Step 2

### Timing Information

Years	Days	Step Size (Days)
0.273785E-04	0.100000E-01	0.500000E-02
Cpu Sec for Time Step = 0.3274E-02 Current Total = 0.2175E-01		

### Equation Performance

Number of N-R Iterations: 1  
 Avg # of Linear Equation Solver Iterations: 3.0  
 Number of Active Nodes: 9.  
 Total Number of Newton-Raphson Iterations: 2  
 Largest Residuals  
 EQ1 R= 0.3765E-08 node= 5 x=0.2500 y=0.2500 z= 1.000

Global Mass & Energy Balances  
Total mass in system at this time: 0.000000E+00 kg  
Total mass of steam in system at this time: 0.000000E+00 kg  
Total enthalpy in system at this time: 0.104781E+03 MJ

Water discharge this time step: 0.000000E+00 kg  
Water input this time step: 0.000000E+00 kg  
Total water discharge: 0.000000E+00 kg  
Total water input: 0.000000E+00 kg

Enthalpy discharge this time step: 0.342068E+00 MJ  
Enthalpy input this time step: 0.000000E+00 MJ  
Total enthalpy discharge: 0.301221E+02 MJ  
Total enthalpy input: 0.301221E+02 MJ

Net kg water discharge (total out-total in): 0.000000E+00  
Net MJ discharge (total out-total in): 0.000000E+00  
Conservation Errors: 0.000000E+00 (mass), -0.100323E+01  
(energy)

Time Step 3

.

.

.

#### 4.1.4.6 Messages

Fatal error conditions and messages for the **fehm** program are given in Table 20. Warning messages (nonfatal errors or informational messages) are given in Table 21.

## DRAFT 4/97

**Table 20. Error conditions that result in program termination**

Error condition	Error message
I/O file error	
Unable to open I/O file	**** Error opening file <i>fileid</i> **** ◊ ◊ ◊ ◊ ****-----**** **** JOB STOPPED **** ****-----****
Coefficient storage file not found	program terminated because coefficient storage file not found
Optional rlp file not found	relative perm file does not exist: stopping
Optional input file not found	ERROR nonexistant file <i>filename</i> STOPPED trying to use optional input file
Unable to open optional input file	ERROR opening <i>filename</i> STOPPED trying to use optional input file
Unable to determine file prefix for AVS output files	FILE ERROR: nmfil2 file: <i>filename</i> unable to determine contour file prefix
Input deck errors	
Coordinate or element data not found	**** COOR Required Input **** -or- **** ELEM Required Input **** ****-----**** **** JOB STOPPED **** ****-----****
Invalid macro read	**** error in input deck : <i>char</i> ****
Invalid AVS keyword read for macro <b>cont</b>	ERROR:READ_AVIS_IO unexpected/invalid character string <i>string</i>
.	.
.	.
.	.
Invalid parameter values (macros using loop construct)	Terminate program execution Fatal error - for array number <i>arraynum</i> macro - <i>macro</i> Group number - <i>groupnum</i> Something other than a real or integer has been specified -or- Line number - <i>line</i> Bad input, check this line
Invalid tracer input	** Using Old Input Enter Temperature Dependency Model Number: 1 - Van Hoff 2 - awwa model, see manual for details **

**DRAFT 4/97**

**Table 20. Error conditions that result in program termination (continued)**

Error condition	Error message
Invalid transport conditions	Fatal error. You specified a Henrys Law species with initial concentrations input for the vapor phase (icns = -2), yet the Henrys Constant is computed as 0 for species number <i>speciesnum</i> and node number <i>nodenum</i> . If you want to simulate a vapor-borne species with no interphase transport, then you must specify a gaseous species (icns = -1).  ERROR -- Illegal Flag to concadiff Code Aborted in concadiff
Optional input file contains data for wrong macro	ERROR --> Macro name in file for macro <i>macroname</i> is <i>wrong_macroname</i> STOPPED trying to use optional input file
Invalid parameters set	
Dual porosity	**** check fracture volumes, stopping **** **** check equivalent continuum VGs ****
Noncondensible gas	cannot input ngas temp in single phase -or- ngas pressure lt 0 at temp and total press given max allowable temperature <i>temp</i> -or- ngas pressure gt total pressure i= <i>i</i> -or- ngas pressure lt 0.
Particle tracking	ERROR: Pcnск in ptrk must be either always positive or always negative. Code aborted in set_ptrk.f
Tracer	ERROR: Can not have both particle tracking (ptrk) and tracer input (trac). Code Aborted in concen.f
Insufficient storage	
Geometric coefficients	program terminated because of insufficient storage
Dual porosity	***** n > n0,stopping *****
Too many negative volumes or finite-element coefficients	too many negative volumes: stopping -or- too many negative coefficients : stopping
Unable to compute local coordinates	iteration in zone did not converge, izonе = <i>zone</i> please check icnl in macro CTRL
Singular matrix in LU decomposition	singular matrix in ludcmp
Solution failed to converge	timestep less than daymin <i>timestep_number current_timestep_size current_simulation_time</i> -or- Tracer Time Step Smaller Than Minimum Step Stop in resetrc

## DRAFT 4/97

**Table 21. Warning conditions and messages**

Warning condition	Message
Thermodynamic variables out of bounds (Note: The code will cut the time step until it can successfully finish a time step. Halving of the time step could eventually result in code termination if the solution fails to converge.)	out of bounds:node <i>node_number</i> <i>p= current_fluid_pressure t= current_temperature</i> -or- <i>p= current_fluid_pressure s= current_saturation</i>
Improper parameter set for reacting species	Warning rate_power is not zero for solid species -or- You must specify either 1 or -1 for h_mult. Check your input. Default is to let the liquid in Henrys law react -or- ERROR: FEHM cannot simulate a solid-liquid-vapor rxn -or- No species have been specified for rxn number <i>irxn</i>
Negative volume or finite-element coefficient	warning : negative volume at node <i>i</i> volume = <i>sx1(i)</i> maximum volume = <i>vol_max</i> -or- warning : negative fe coefficient at node <i>i</i> neighbor node <i>ncon(j)</i> coeff sum = <i>sumsx</i> maximum area divided by length = <i>vol_max</i>
Error writing AVS output	
Error opening file	OPEN ERROR: <i>filename</i>
Binary geometry file	WRITE ERROR: geo <i>geometry_file</i>
Binary header file	CLOSE ERROR: geo <i>geometry_file</i>
Binary node file	WRITE ERROR: binary header <i>header_file_type</i>
ASCII geometry file (wrong format specified)	CLOSE ERROR: binary header <i>header_file_type</i> WRITE ERROR: binary node <i>node_file_type</i> CLOSE ERROR: binary node <i>node_file_type</i> AVS_WRITE_CORD Unformatted IO not implemented No action AVS_WRITE_STRUC Unformatted IO not implemented No action
ASCII geometry file (invalid output format)	AVS_WRITE_CORD Invalid output format No action AVS_WRITE_STRUC Invalid output format No action
ASCII scalar node file	----- ERROR: WRITE_AVN_NODE_S <i>nscalar = scalars_requested</i> is greater than <i>maxscalar = max_number_of_scalars</i> Subroutine only able to handle up to 8 scalars -----

**DRAFT 4/97**

**Table 21. Warning conditions and messages (continued)**

Warning condition	Message
ASCII scalar node file (continued)	<p>-----</p> <p>WARNING: WRITE_AVN_NODE_S Pressure defined, but not Vapor or Liquid There will be no AVS output for scalar pressure</p> <p>-----</p>
	<p>-----</p> <p>WARNING: WRITE_AVN_NODE_S Vapor defined, but not pressure There will be no AVS output for scalar pressure</p> <p>-----</p>
	<p>-----</p> <p>WARNING: WRITE_AVN_NODE_S Liquid defined, but not pressure There will be no AVS output for scalar pressure</p> <p>-----</p>
	<p>ERROR: WRITE ASCII NODE S: Unknown IO combination for <i>number</i> scalars pressure, temperature, vapor, saturation, and liquid</p> <p>-----</p>
BINARY scalar node file	<p>ERROR: WRITE_BINARY_NODE_V <i>nvector</i> = <i>nvec</i> is greater than <i>maxvector</i> = <i>MAXVECTOR</i></p> <p>-----</p> <p>ERROR: WRITE_BINARY_NODE_V Unknown vector value <i>nvec</i> Number beyond bounds: min to max Occurred using <i>pn</i>* <i>i</i> for vector 1 -&gt; <i>pn</i>* * is xl, xv, yl, yv, zl, or zv Number beyond bounds: min to max Occurred using * <i>i</i> for material <i>n</i> -&gt; <i>th</i>* * is pnx, pny, pnz, thx, thy, or thz</p> <p>ERROR: WRITE_BINARY_NODE_S <i>nscalar</i> = <i>nscalar</i> <i>nmaxscalar</i> = <i>MAXSCALAR</i></p> <p>-----</p>
	<p>WARNING: WRITE_BINARY_NODE_S Pressure defined, but not Vapor or Liquid There will be no AVS output for scalar pressure</p> <p>-----</p>
	<p>WARNING: WRITE_BINARY_NODE_S Vapor defined, but not pressure There will be no AVS output for scalar pressure</p> <p>-----</p>
	<p>WARNING: WRITE_BINARY_NODE_S Liquid defined, but not pressure There will be no AVS output for scalar pressure</p> <p>-----</p>

**DRAFT 4/97****Table 21. Warning conditions and messages (continued)**

Warning condition	Message
BINARY scalar node file (continued)	ERROR: WRITE_BINARY_NODE_S: Unknown IO combination for <i>number</i> scalars pressure, temperature, vapor, saturation, and liquid ERROR: WRITE_BINARY_NODE_S Unknown scalar value <i>nscalar</i> Number beyond bounds: min to max Occurred using * i for ? -> pn* * scalar parameter, ? description ERROR: WRITE_BINARY_NODE_CON <i>nspeci</i> = <i>nspeci</i> is greater than max con =

**4.1.4.7 Other input/output**

N/A

**DRAFT 4/97**